REPORT DOCUMENTATION PAGE

Form Approved
OMB NO. 0704-0188

1. AGENCY USE ONLY (Leave Blank)

2. REPORT DATE 02.14.02

3. REPORT TYPE AND DATES COVERED
Final, : 05/15/98 - 11/14/01

4. TITLE AND SUBTITLE
Computer Aided Design Tools for Mixed Electronic/Photonic VLSI

5. FUNDING NUMBERS
DAAG55-98-0319

6. AUTHOR(S)
Michael S. Shur

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
Rensselaer Polytechnic Institute

8. PERFORMING ORGANIZATION REPORT NUMBER Final

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)

U. S. Army Research Office
P.O. Box 12211
Research Triangle Park, NC 27709-2211

10. SPONSORING / MONITORING AGENCY REPORT NUMBER
38879.2-EL

11. SUPPLEMENTARY NOTES
The views, opinions and/or findings contained in this report are those of the author(s) and should not be construed as an official Department of the Army position, policy or decision, unless so designated by other documentation.

12 a. DISTRIBUTION / AVAILABILITY STATEMENT

Approved for public release; distribution unlimited.

12 b. DISTRIBUTION CODE

13. ABSTRACT (Maximum 200 words)

In this project, we have developed the physics-based models for photonic elements (sources, detectors and interconnects), expressed these in terms of electrical equivalents, and implemented them in the new Photonic SPICE. This transformation between the optical and the electrical dimensions have been performed in such a way that all effects of optical emission, propagation and detection are reproduced by their electronic counterparts. The physics-based models have been checked against experimental results.

14. SUBJECT TERMS
SPICE
Photonic
VLSI

15. NUMBER OF PAGES
6

16. PRICE CODE

17. SECURITY CLASSIFICATION OR REPORT	18. SECURITY CLASSIFICATION ON THIS PAGE	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	UL

NSN 7540-01-280-5500

Standard Form 298 (Rev.2-89)
Prescribed by ANSI Std. 239-18
298-102

REPORT DOCUMENTATION PAGE (SF298)
(Continuation Sheet)

REPORT DOCUMENTATION PAGE (SF298)
(Continuation Sheets)

Computer Aided Design Tools for Mixed Electronic/Photonic VLSI

The main program goals of this project were to develop the physics-based models for photonic elements (sources, detectors and interconnects), to express these in terms of electrical equivalents, and to implement them in the new Photonic SPICE. This transformation between the optical and the electrical dimensions were performed in such a way that all effects of optical emission, propagation and detection were reproduced by their electronic counterparts. The physics-based models were checked against experimental results.

We performed modeling of small-signal and transient effects in vertical cavity surface emitting lasers (VCSELs), and the modeling of optical detectors and interconnects. The models were tested against experiments and implemented in Photonic SPICE.

A VCSEL continuous wave model that includes self-heating was developed. The model reproduces all the features of available experimental data. The equivalent circuit of the model, implemented in Photonic SPICE, consists of interacting electrical, optical and thermal subcircuits.

In the design of VLSI systems, computer-aided design (CAD) tools have become a prerequisite for assuring correct operation and optimized solutions. Without these tools, multiple design and fabrication iterations are required. As the momentum for the development of mixed photonic/electronic systems are steadily increasing, it is of great importance to establish a CAD infrastructure for effective and accurate modeling and simulation of such heterogeneous systems.

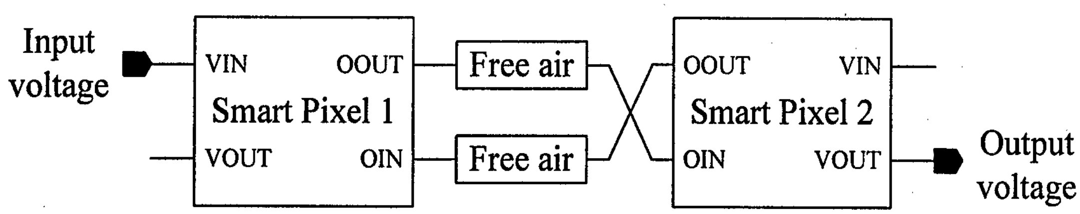
Our approach to modeling and simulation of heterogeneous photonic/ electronic systems took advantage of the fact that the ubiquitous SPICE simulator can be viewed as a general solver for a set of ordinary differential equations. Even though SPICE does all its calculations in units of volts and amperes, there are, of course, no restrictions on how to interpret the results. Hence, the total circuit can be divided into several physical domains, each having a separate set of physical quantities as unknowns. Our simulator, AIM-Spice [1], currently supports three domains: electrical, optical and thermal. The advantages of this approach are its inherent efficiency and flexibility. All domains are simulated in a single kernel running in a single process. Thus, there are no overhead due to inter process and inter kernel communications. The approach is flexible since a single device can span more than one domain. This is particularly important in the modeling and simulation of optoelectronic interconnects, where a single device has ports in both the electrical and the optical domains.

In the report, we first give an overview of the models that are implemented. Then we describe a test case where we have used our simulator to study a smart-pixel based system for free-space optical interconnects. Finally, we summarize the conclusions of our study.

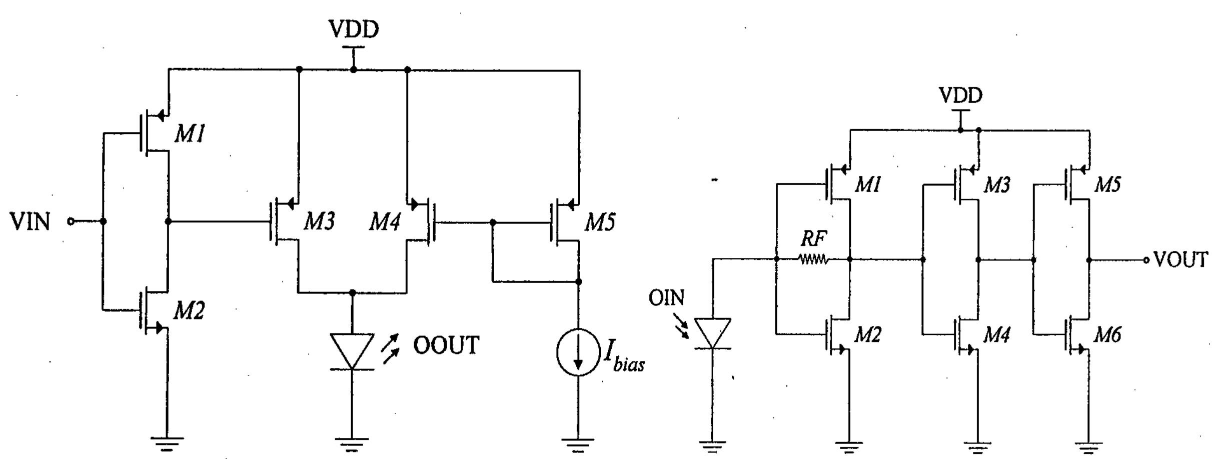
Device Models: Important devices used in optoelectronic systems are laser diodes and light-emitting-diodes (LEDs) on the transmitter side. On the receiver side, *p-i-n* diodes, Schottky barrier diodes, and metal-semiconductor-metal devices are commonly used detectors.

Among the different emitter structures listed above, the vertical cavity surface emitting laser (VCSEL) has stood out as the best candidate for many applications (see, for example, [2]). A VCSEL model implemented in AIM-Spice is based on first-order rate equations and includes effects such as self-heating and parasitic resistances and leakage currents. On the receiver end, we have implemented models for both *p-i-n* diodes and metal-semiconductor-metal (MSM) structures. The models are described in detail in [3,4].

Case Study–An optical Interconnect System: Smart-pixel arrays (SMAs) have been introduced as a viable technology for free-space optical interconnects [5]. A cross sectional view of a single pixel in a SMA is shown in Fig. 1. Note that each pixel consists of a VCSEL and an MSM photodetector to provide bi-directional data flow. The other elements of the structure are the CMOS integrated circuit on a Si substrate, the photonic devices on a GaAs substrate and the micro lenses on a glass substrate. The elements are stacked together using a special process.



(a)



(b)

Fig. 1. Cross sectional view of a single pixel in a SMA.

a) Block diagram; b) Transistor diagram of CMOS driver and receiver.

To illustrate how this smart pixel optical interconnect system can be simulated using our single kernel simulator, we connect two smart-pixels back-to-back having free air in between. The block diagram of this setup is shown in Fig. 1a. To obtain realistic simulation results, we have extracted VCSEL and MSM parameters from real devices fabricated at Honeywell Corporation. Furthermore, for the CMOS circuits, we used BSIM3v3.1 parameter sets for a commercial quarter-micron process running at a power supply level of 2.5V. The schematics of the CMOS driver and the receiver are shown in Fig. 1 b. To include the interconnect effect, we included parasitic capacitances and inductors for both the driver-VCSEL and the receiver-MSM interconnects. The free air medium (see the block diagram in Fig. 2 a) was modeled in terms of a simple attenuation. The top level SPICE description of the system shown in Fig. 2a is listed below. The subcircuit definition of the smart pixel is also included in the listing. Note that there are both electrical and optical nodes in the circuit.

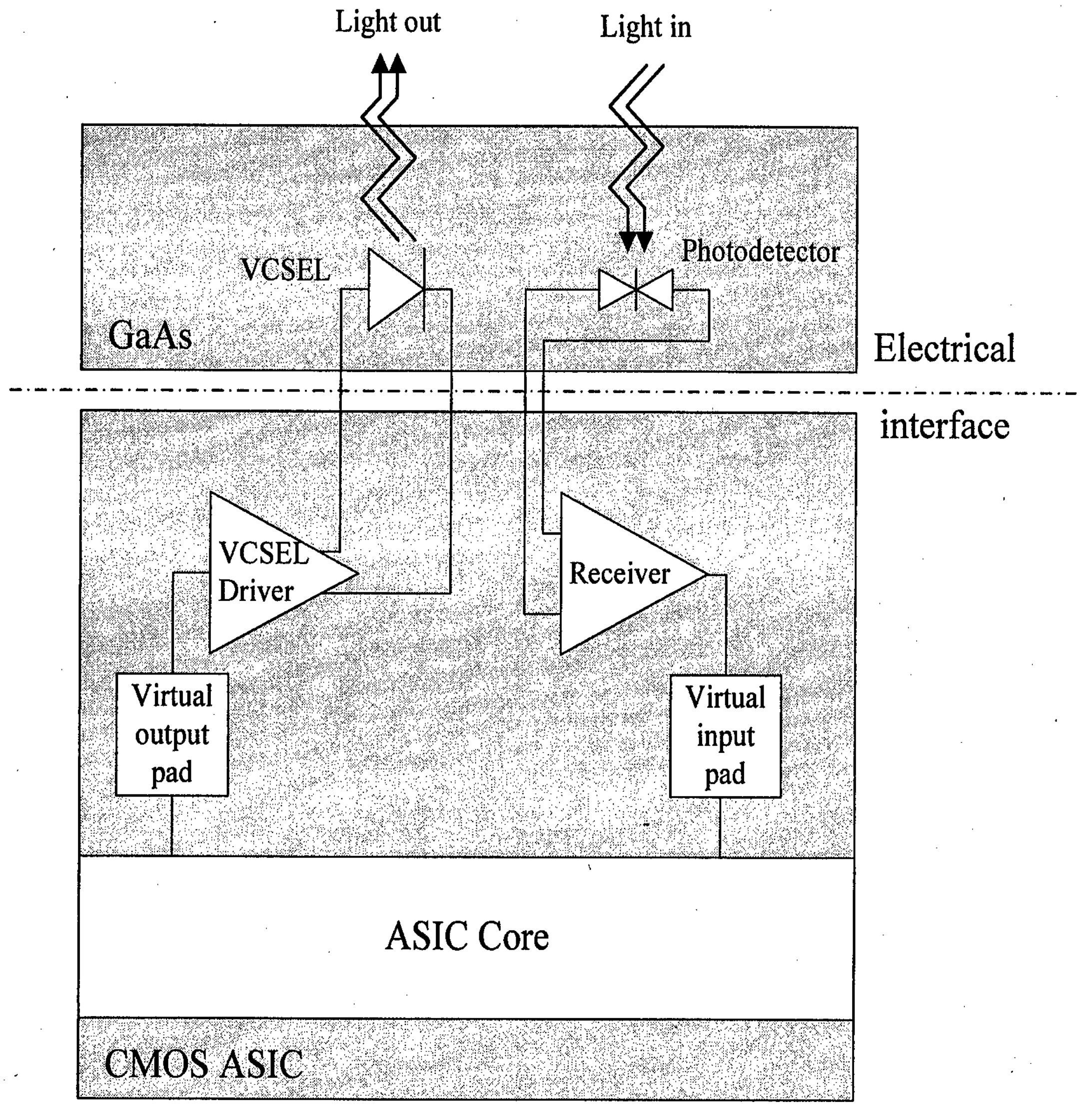


Fig. 2. Schematics of a free air interconnect system

```
Free-air optical interconnect

.lib mos_models.cir
.lib vcsel_model.cir
.lib msm_model.cir

* Power supplies
vdd1 vdd1 0 dc 2.5
vss1 vss1 0 dc 0
vdd2 vdd2 0 dc 2.5
vss2 vss2 0 dc 0

* Input signal generator
xsg vdd1 vss1 vin signal_generator

* Two smart pixels connected back-to-bak
xp1 vdd1 vss1 vin vout1 oin1 oout1 smart_pixel
xp2 vdd2 vss2 vin2 vout oin2 oout2 smart_pixel
xtr1 oout1 oin2 free_air
xtr2 oout2 oin1 free_air

* The smart pixel subcircuit
.subckt smart_pixel vdd vss vin vout oin oout
xdriver vdd vss vin oout driver
xreceiver vdd vss oin vout receiver
.ends smart_pixel
```

The simulation results for a simple 1GHz clock signal used as the input signal waveform is shown in Fig. 3. We note that the output waveform is delayed about 0.5 ns though the link. The estimated maximum frequency performance of this circuit is about 1.2 GHz. This frequency can be increased by utilizing more sophisticated analog CMOS circuitry and signaling protocols (for example low-voltage differential signaling (LVDS)).

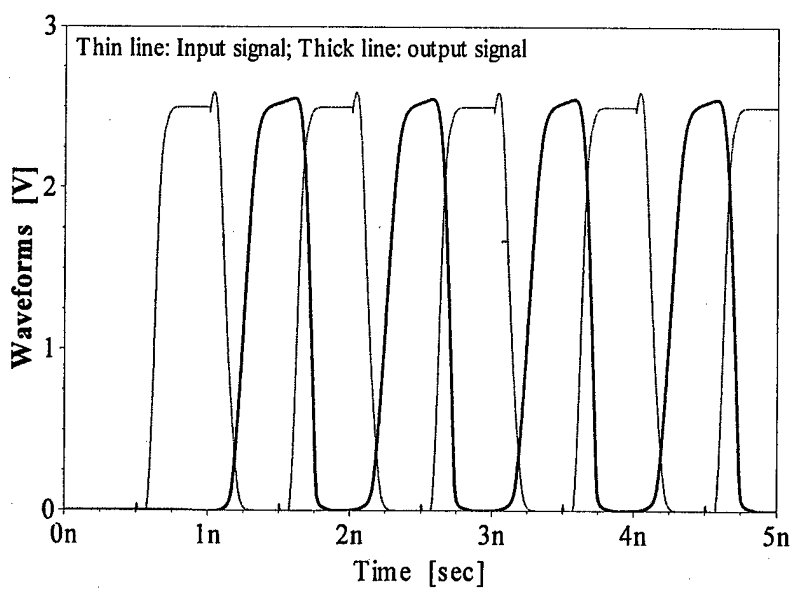


Fig. 3. Computed waveforms

In summary, we have described an efficient approach to the modeling and simulation of mixed-domain electrical/optical systems utilizing a single kernel simulator. We illustrated the approach by presenting a case study of an optoelectronic interconnect system.

References

1 The AIM-Spice Web site at http://www.aimspice.com.

2 D.A.B. Miller, "Rationale and Challenges for Optical Interconnects to Electronic Chips", *Proceedings of the IEEE*, vol. 88, no. 6, pp. 728-749, June 2000.

3 J. Deng, M.S. Shur, T.A. Fjeldly, S. Baier, "SPICE Modeling for Mixed Electronic/Photonic VLSI", presented at the *XXVIth General Assembly of the International Union of Radio Science*, Toronto, Canada, Aug. 1999.

4 J. Deng, M. S. Shur, T. A. Fjeldly, S. Baier, "CAD Tools and Optical Device Modeling for Mixed Electronic/Photonic VLSI", International Journal of High Speed Electronics and Systems, vol. 10, no. 1, pp. 299-308, 2000.

5 Y. Liu, E.M. Strzelecka, J. Nohava, M.K. Hibbs-Brenner, and E. Towe, "Smart-Pixel Array Technology for Free-Space Optical Interconnects", *Proceedings of the IEEE*, vol. 88, no. 6, pp. 764-768, June 2000

The results of this work have been published in the following papers:

1. T. Ytterdal, T. A. Fjeldly, S. Baier, J. Deng, and M. S. Shur, Modeling and Simulation of Optoelectronic Interconnect Systems Using a Single Kernel Simulator, in Proceedings of ICCDCS-2002, submitted for publication
2. J. Deng, M. S. Shur, T. A. Fjeldly, S. Baier and T. Ytterdal; SPICE Modeling for Mixed Electronic/Photonic VLSI, paper 8.6, Rensselaer Polytechnic Institute, presented at TECHCON 2000 on 21-Sep-2000 (Pub P000740)
3. T. Ytterdal, T. A. Fjeldly, M. S. Shur, Beyond SPICE, A Review of Modern Analog Circuit Simulation Techniques, in "Silicon and Beyond. Advanced Device Models and circuit simulators, M. S. Shur and T. A. Fjeldly, Editors, World Scientific, 2000, pp. 113-136
4. J. Deng, M. S. Shur, T. A. Fjeldly, S. Baier, CAD Tools and Optical Device Models for Mixed Electronic/Photonic VLSI, International Journal of High Speed Electronics and Systems, International Journal of High Speed Electronics and Systems, Invited, Volume 10, No 1, pp. 299-308, March 2000
5. M. S. Shur and T. A. Fjeldly, Editors, Silicon and Beyond: Advanced Device Models and Circuit Simulators (Selected Topics in Electronics and Systems), World Scientific Pub Co; ISBN: 9810242808, April 2000